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CREATION FORGING DIES' FUNCTIONAL SURFACE USING PRESENT CAD/CAM SYSTEM

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ABSTRACT: *Contribution deals with methods for creating functional surfaces of forming dies using CAD systems and focuses to bulk forming processes - die forging. The source for functional surfaces design of forging die is solid model of forged piece with defined mould joint, chamfers and radii. Next are presented two methods of forging dies creating according to CAD systems possibilities - mould cavity mode and assembly mode (Merge-Cut Out command). Both method are presented step by step and there are compared their advantages from the user's view.*
KEYWORDS: *CAD, functional surface, forging die, mould cavity mode, Merge-Cut Out*

INTRODUCTION

The forming technology represents shape-changing technological operation by which a forming die acts to a material to produce solid or hollow bodies from solid or flat raw products – forgings, pressings, stampings etc. Parts produced by forming processes represent simple or complicated in shape parts and its design uses complicated in shape surfaces too. The significant forming die characteristic is a singularity - each part's shape could be realized by different technological procedures, hence by the different concepts in design of forming dies. Parts produced by forming are made at a single movement of complicated in shape forming dies within a very short production time. [1]

Differences in shape of parts produced by forming in a large extent follow from used shape of incoming raw product.

Cold forming (stamping) operations process flat raw products (sheets) and there is no significant change in thickness but relevant change in shape by plastic deformation.

Hot (bulk) forming operations process bars, rods and sheets by large volume plastic deformation and redistribution of raw product volumes and sections. This shape difference of parts produced by cold and hot forming operations needs to be taken into consideration by die designer in CAD design of forming die functional surfaces and in the choice of techniques by which forming dies surfaces will be created as well. [2]

SURFACES USED IN CAD/CAM SYSTEMS

The CAD and CAD/CAM software packages focused to 3D modelling and design show a significant role in design of forming dies at present. Its improving led to modelling techniques evolution as well. Complicated mathematic description of functional surfaces was implemented into CAD and CAD/CAM software packages and it allows designing complex free form surfaces with a high degree of geometric constraint applied that are typical for outer and inner car's body components. [3]

There are used two types of surfaces in CAD/CAM systems primarily at present: [3,4,5,6,7].

1. **Regular (or canonical) surfaces** - include revolved surfaces, such as cylinders, cones, spheres, and tori, and extruded surfaces (linear in one direction). These surfaces are usually created using REVOLVE and EXTRUDE commands and both closed section (or sketch) and path of extrusion or axis of revolution are necessary to define. As the primary surfaces are created, other commands can be applied to create rounds, chamfers, ribs, drafts etc. The main attribute of these surfaces is exact definition of sketch described by dimensions of lines, radii, arcs, polygons, etc.

2. **Freeform surfaces** - represents more complex shapes of surfaces. They do not have rigid radial dimensions, unlike regular surfaces such as planes, cylinders and conic surfaces. Most systems use non-uniform rational B-spline (NURBS) mathematics to describe the surface forms. The forms of freeform surfaces (and curves) are not stored or defined in CAD software in terms of polynomial equations, but by their poles, degree, and number of patches (segments with spline curves). CAD software packages use two basic methods for the creation of freeform surfaces:

   the first begins with construction curves (splines) from which the 3D surface is then swept (section along guide rail), or meshed (lofted) through.

   the second method is direct creation of the surface with manipulation of the surface poles/control points.

From these initially created surfaces, other surfaces are constructed using either derived methods such as

offset or angled extensions from surfaces; or via bridging and blending between groups of surfaces.

CREATION OF FORGING DIE CAVITY

In forming dies design with computer aid is commonly used principle of die functional surfaces creation based on solid or surface model from which functional surface of die is derived. It is useful to note die functional surface is negative to part surface. This procedure is typical for cavity creation of moulds and forging dies as well as functional surfaces of stamping dies in automotive industry. The main advantage of mentioned is derivation of die surface directly from part surface and elimination of faults at die model creation from 2D drawing. Next are presented two methods for creation of forging die cavity as its functional surface using CAD/CAM software Pro/Engineer: the first one is based on Pro/MOLD module (mould cavity mode) and the second one is based on merging positioned models by Cut Out operation (assembly mode). [8,9]

Forged part

Part produced by forging and machined to desired dimensions is shown in Figure 1. The part is created by flanged head and shaft. Flange is symmetrical with shaped noses and threaded holes. Shaft is created by two cylindrical diameters with G-neck at flange to shaft transition and relive for snap-ring at the end. The flange is relieved with sphere cut.
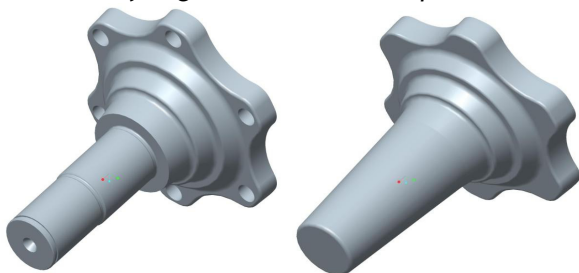

Figure 1. Final part   Figure 2. Forging shape

Considering technological design of part, required accuracy of dimensions and surface quality given by 2D drawing was proposed machining allowances, dividing plane, bevels and radii of sharp edges. Final shape of forging is shown at Figure 2.

Due to complex shape of forging it isn't possible to make it at single forging operation. Technological process of forging was proposed according to required calculations at which dimensions of raw product, operation steps, outflow groove and forces were calculated. Intermediate operations aren't considered here, only final forging operation.

Creation of forging die functional surfaces in Pro/MOLD

CAD/CAM software Pro/Engineer ver. Wildfire 5.0 was used to create forging die cavity. The software Pro/Engineer continues after rebranding at present as Creo 2.0 software package with preserved all functionalities of key properties and modular structure as well. Pro/MOLD represents one of optional modules of software package for cavities creation of casting moulds, injection moulds for plastics and forging dies as well as its components (runners, inserts, sand cores, etc.). After module starts and file creation working area and icons are shown in right toolbar. Based on the process

controlled approach to cavity creation icons are aligned in sequence of commands from up to down. When dies cavity are created designer uses commands step by step in order of icons.

The basic step at cavity creation using "mould cavity mode" is reference part creation. The reference part is then used in software module to create die cavity. The part created in basic module of software is loaded to Pro/MOLD module as REFERENCE MODEL. The reference model is one way associated (all changes in part model are respected to reference model, but changes in reference model aren't respected to part model).
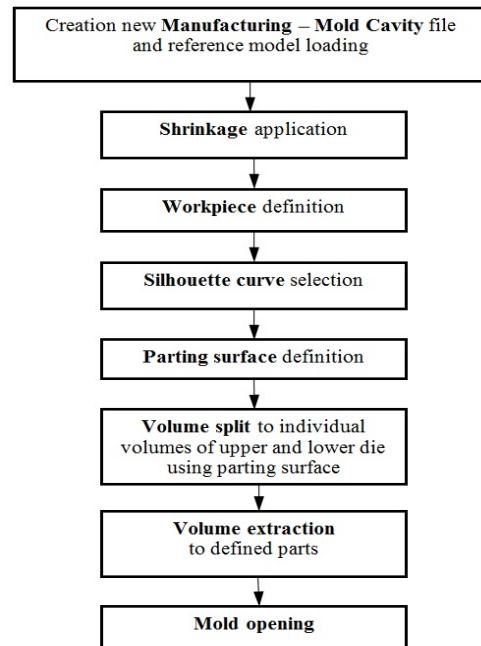

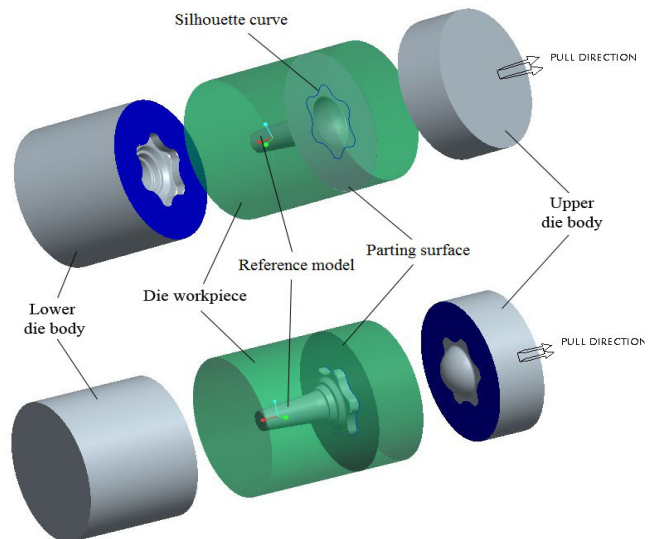Figure 3. Flowchart at die cavity creation in module Pro/MOLD (mold cavity mode)

Figure 4. Models of upper and lower forging die created as mould cavity mode

Figure 3 shows other steps to die cavity creation by "mould cavity mode" method. The second step defines Shrinkage of reference model due to forging temperature. Shrinkage can be applied in two ways: to whole part according to its coordinate system or separately to each dimension. Workpiece around reference model is defined in the third step. The workpiece represents outer shape of die created.

Depending on reference model and workpiece position are then defined dividing planes and each volumes of created die. The workpiece is then divided by dividing planes and volumes onto required number of die parts that are extracted from workpiece to separate parts. These parts then create upper and lower part of dies for forging. Final shape of upper and lower part of dies and names of each features are shown in Figure 4.

Creation of forging die functional surfaces in Assembly mode

The way of die cavity creation in assembly mode requires 3D models (solid or surface) of forging and workpieces of upper and lower die parts. The basic step is positioning of forging and workpieces models in assembly mode of software package Pro/Engineer relative to parting surface position.
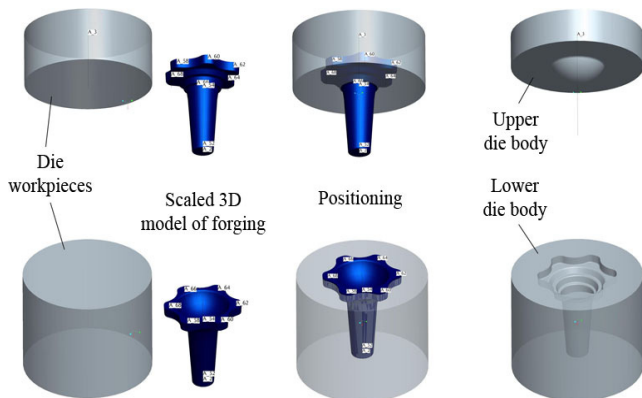

Figure 5. Models of upper/lower forging dies created as assembly mode
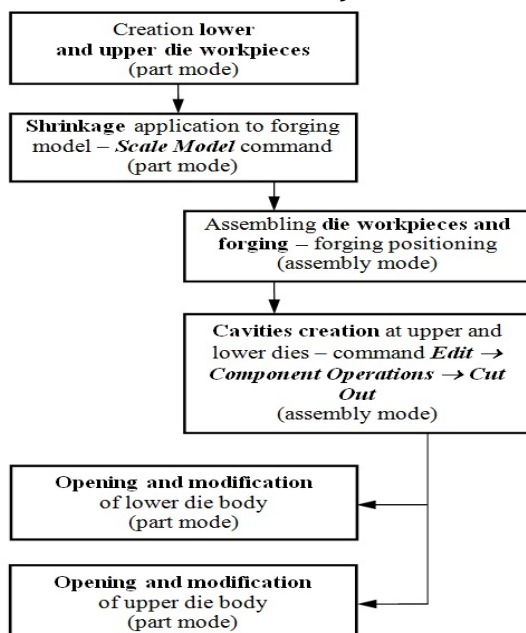

Figure 6. Flowchart at die cavity creation by Cut Out Component Operation (assembly mode)

Figure 5 shows 3D models of lower die workpiece and forging. 3D model of forging is necessary to scale according to shrinkage percentage (command Edit → Scale Model). Forging and lower die workpiece position need to be fully constrained using standard assembly commands (Mate, Align, Insert, ...). Cavity in lower die is then created by command Edit → Component Operations → Cut Out with selection of part to perform Cut Out process to at first (lower die

workpiece) and selection of reference parts for Cut Out process at second (3D model of forging with shrinkage applied). Created cavity is then negative to forging 3D model in lower die workpiece – Figure 5. The upper die part is then created by positioning 3D model of forging with shrinkage applied and upper die workpiece and application of command mentioned. Note, when complicated forging parts are processed, closed cavities may occur in lower or upper die workpieces that need to be remodelled (cut out) in part mode.

Figure 6 shows steps to die cavity creation by "assembly mode" method.

Comparison of methods for die cavity creation

Figure 7 and Figure 8 shows and compare process tree at both methods for die cavity creation.

In the case of die cavity creation by mold cavity mode process tree represents complicated structure with items adding by command sequence according to flowchart in Figure 3. The base is reference model, indicated by "_ref" and scaled by shrinkage percentage that is used as core for creation of die cavity. Then workpiece is defined and added to process tree. Next silhouette curve is added and used for parting surface creation - Skirt surface in process tree. The very important is splitting workpiece volume to upper and lower dies and volumes extraction by which are added 3D models of upper and lower dies.

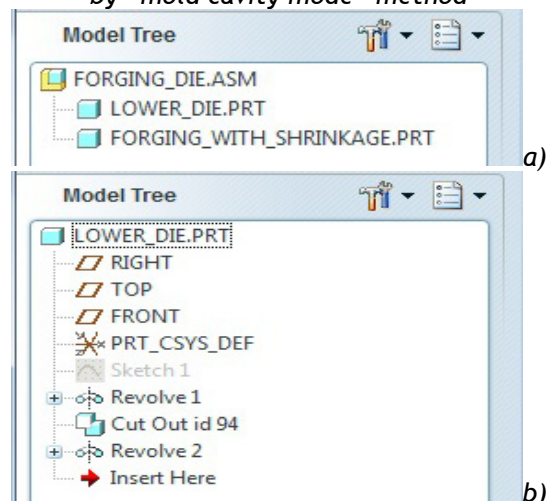*Figure 7. Process tree at cavity creation by "mold cavity mode" method*

Figure 8. Process trees at cavity creation by "assembly mode" method
a) process tree in assembly mode
b) process tree in part mode for lower die

There are two process trees at creation of die cavity by assembly mode: the first process tree is in assembly mode where die workpiece and forging model are positioned and merged by Cut Out operation and the second process tree is in part mode where die workpiece modelling is finished. The cavity is created in assembly mode and process tree is very simple when compared to previous method. After material removing by command Edit → Component Operations → Cut Out die cavity is created in die workpiece and it is added in process tree in part mode after lower die part opening (item Cut Out id). Following operations on lower or upper die are realized in part mode after its opening.

The main advantage of die cavity creation in mold cavity mode is automatic creation of parting surfaces and splitting of workpiece volumes to desired die volumes (upper and lower). This mode could be called technologically oriented approach to die cavity creation. When dies cavity is created in assembly mode, designer must realize each operation step by step manually including positioning of workpiece and forging and creating parting surface by using standard methods and commands allowed in assembly mode.

CONCLUSIONS

Forming dies design is realized mainly in 3D space at present. CAD, resp. CAD/CAM software packages offers die designers powerful tools for creation complicated shapes of parts as well as complicated shapes of functional surfaces of forming dies. The contribution was focused to procedures used for cavity creation at forging dies design process. The main advantage of presented procedures is the associativity that offers automatic transfer of changes realised in 3D model of part, resp. forging to all modules of software that had been used, without any data transformation through neutral file formats. Substantial reduction of design-technology cycle is then reached, possible mistakes at data transformation are reduced and designer's work efficiency is increased as well.

Acknowledgement

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